

Modifying Features

I-DEAS® Tutorials: Fundamental Skills

Learn how to:

- select features directly
- modify feature parameters
- modify dimensions
- modify a feature's wireframe geometry
- modify a section underlying a feature

Before you begin...

Prerequisite tutorials:

1. Introducing the I-DEAS™ Interface

—or—

Quick Tips to Using I-DEAS

—and—

Creating Parts

2. Sketching and Constraining

3. Dimensioning

4. Building Sections

5. Using Sketch Planes and Understanding Sketch Pads

6. Extruding and Revolving Features

7. Adding Features with Associativity

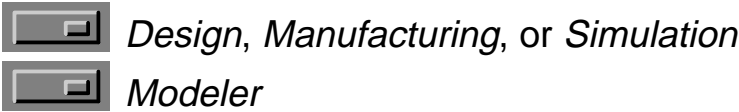
If you didn't start I-DEAS with a new (empty) model file, open a new one now and name it "modify."



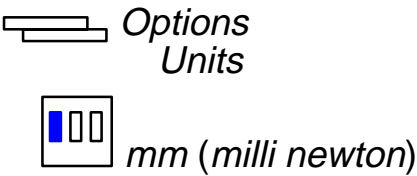
Open Model File form



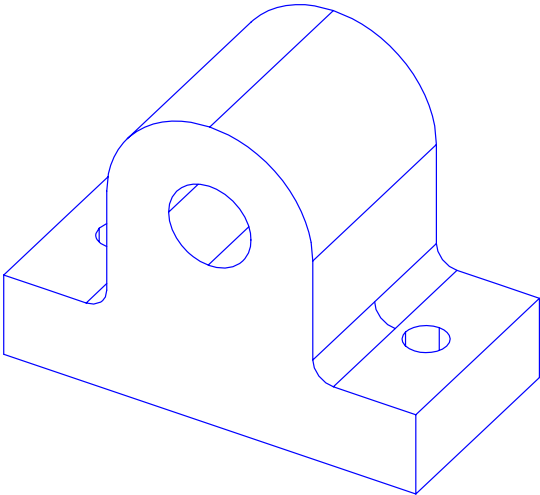
Make sure you're in the following application and task:



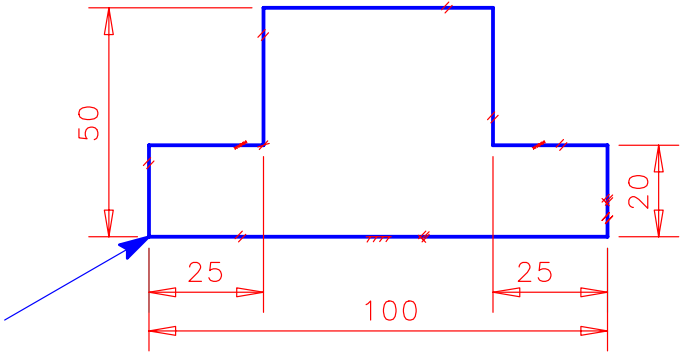
Set your units to mm.



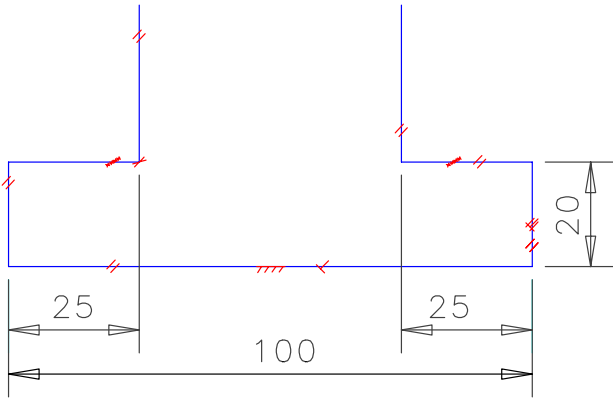
Follow the instructions on the next few pages to create a part similar to the one shown below.



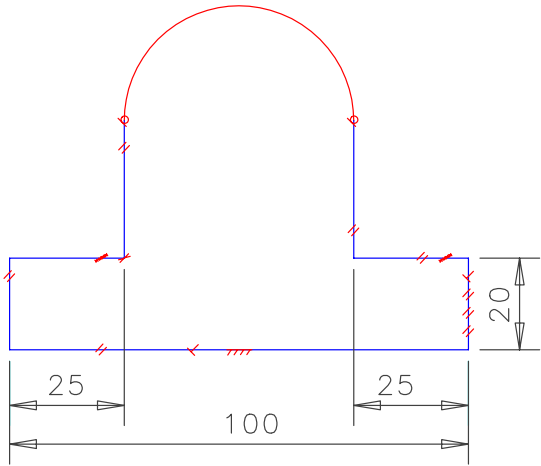
Step 1. Start in the lower left-hand corner and sketch this shape and dimension it as shown.



Step 2. Delete the top line.



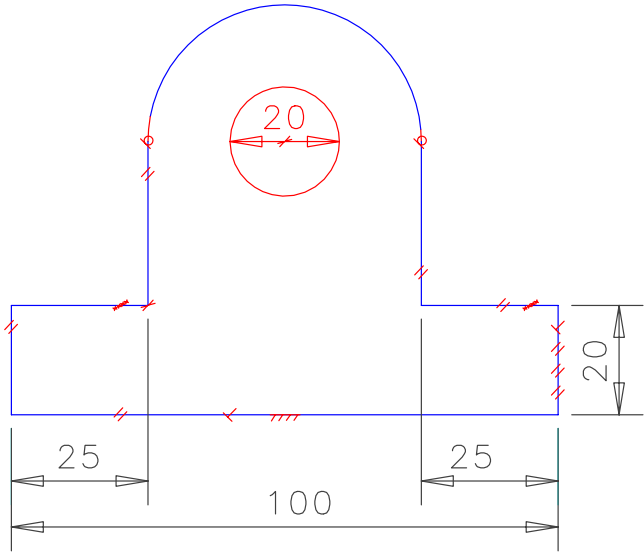
Step 3. Fillet between the two vertical lines. Accept the form defaults.



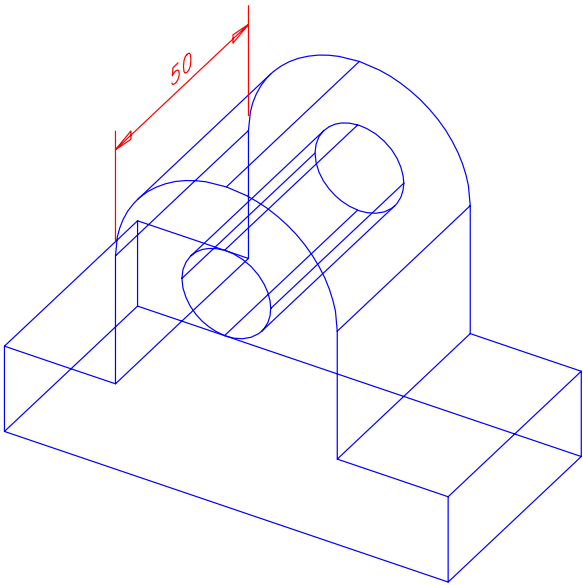
Hint



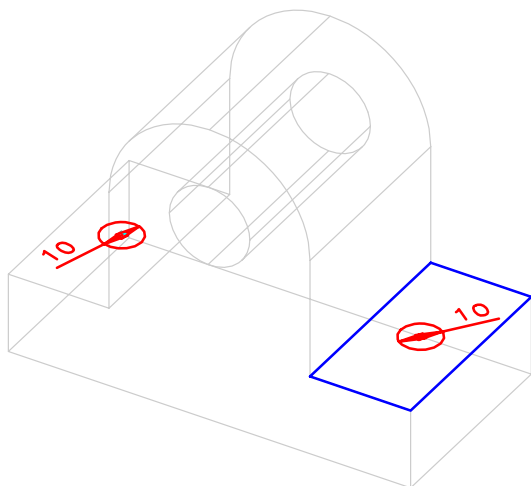
Step 4. Sketch a circle on the center point and modify the dimensions as shown.



Step 5. Extrude the sketch and the circle at the same time (50mm).

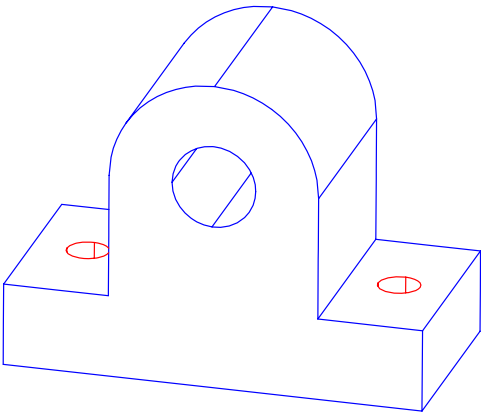


Step 6. Use *Sketch in Place* on either side of the base and sketch two circles. Use the *Dynamic Navigator* to center the circles on the faces. Modify the dimensions as shown.

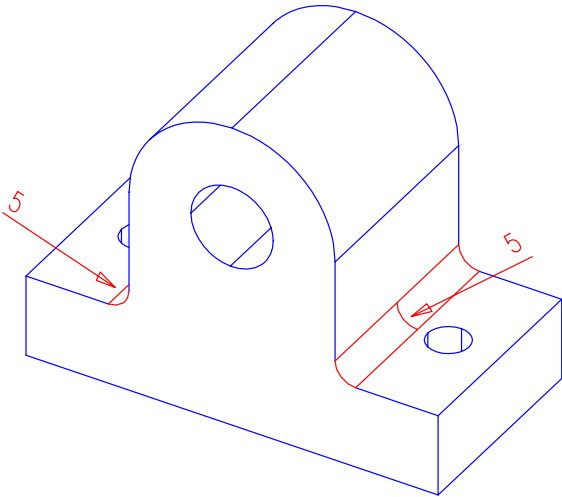


To create both holes as one feature, you must sketch them on the same plane. Do not use a second *Sketch in Place* command for the second circle. To dimension the location of the second circle, you could dimension it to the first circle, or dimension it to other points using the *Focus* option.

Step 7. Cut both holes through the part at the same time.



Step 8. Fillet both sides at the same time.



Hint

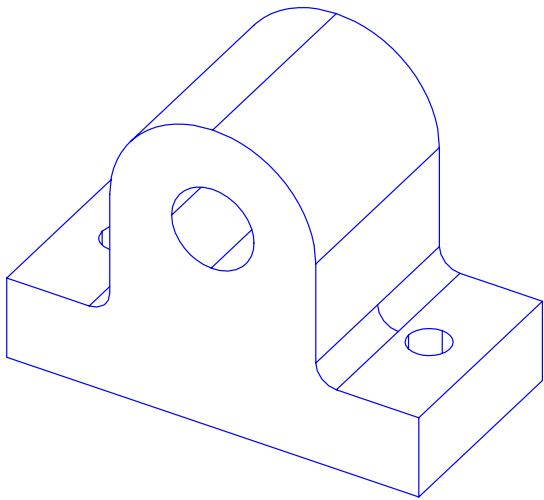


Type 5 in *I-DEAS Prompt* and press Return.



(Done)

Result



Save your model file.



Warning!

If you are prompted by I-DEAS to save your model file, respond:



Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

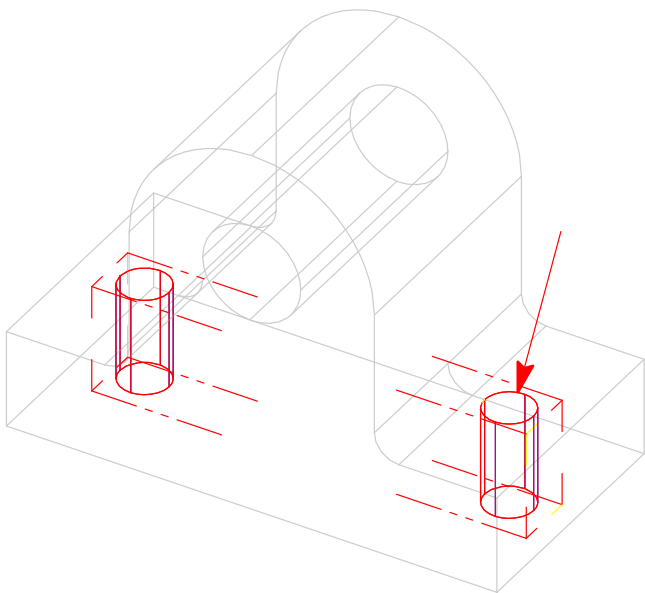
Hint

To reopen your model file to the previous save, press Control-z.

To modify features, you must be able to select them. There are two primary ways to select a feature:

- directly picking the feature
- using *History Access*

This tutorial shows you how to select features directly from the part. The next tutorial presents the techniques for selecting features with history access.



Practice selecting each of the three features by directly picking with no icon active.

It will take three picks at the same location to select a feature (or two picks to pick the part). It is important that you do not move the mouse between picks.

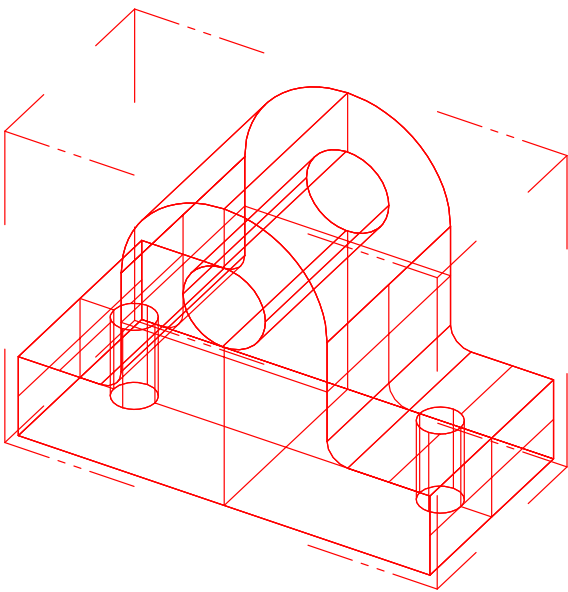


Deselect All

Select the first feature of the part by picking three times on the front face.



pick front face of part



Pick the feature of the two cutouts on the base.



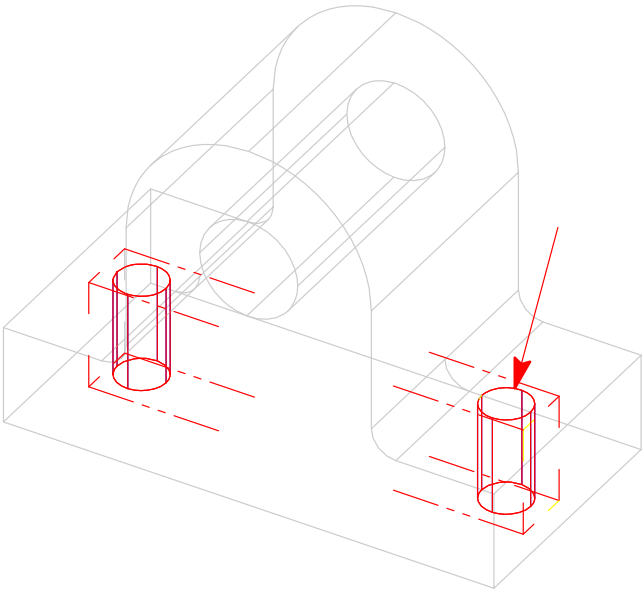
Deselect All

Remember

Do not move the mouse between mouse clicks.



pick on the cutout



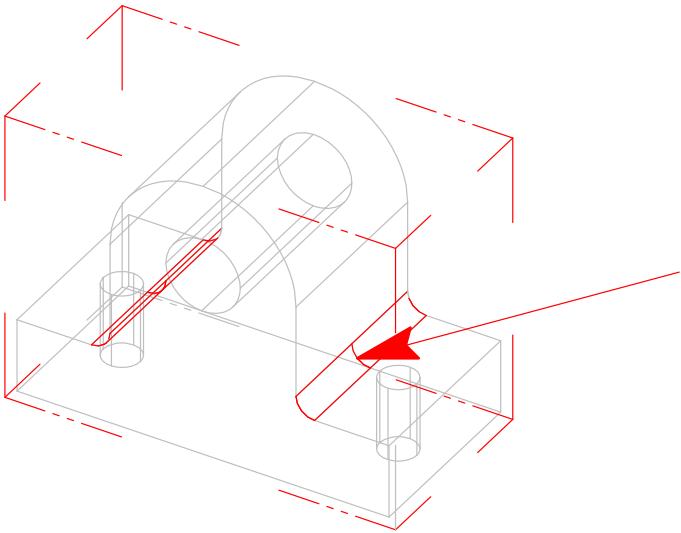
Pick the fillet feature.



Deselect All



pick on the fillet



Deselect All

When you create features with parameters that are entered on a form, you can modify those parameters using the *Feature Parameters* option of the *Modify* command.

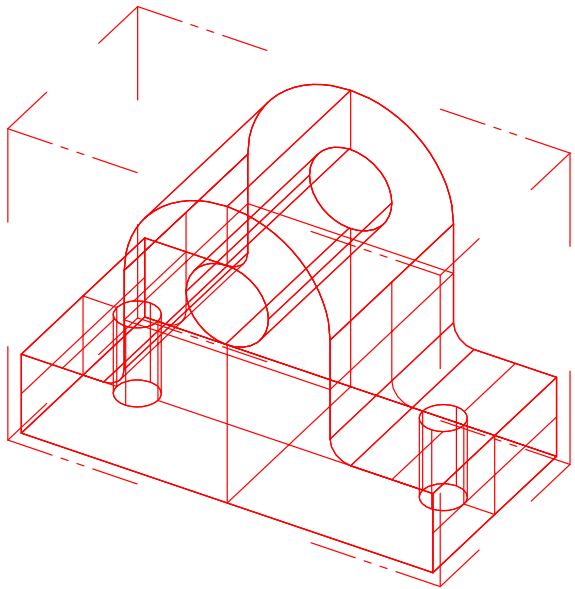
This allows you to change the direction of an extrude, change a protrude into a cutout, or change other options, such as a draft angle.

You cannot, however, change an extrude into a revolve, because this is not a parameter on the form.

Select the first extrusion feature.

Hint

Pick the front face of the part three times.



Modify the extrusion distance to 30.



Feature Parameters

Extrude Section form

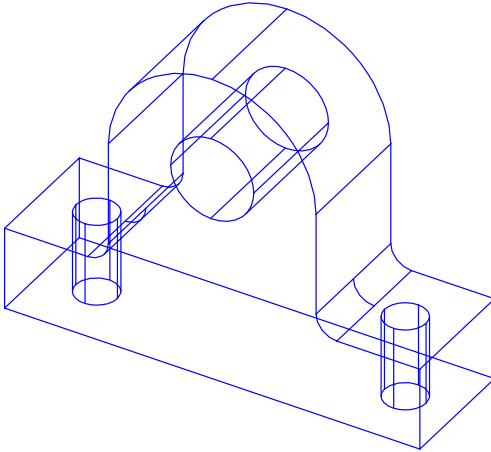


Distance: 30



OK

Update twice. The first time updates the feature, the second time updates the entire part.



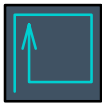
Don't get in the habit of counting the number of mouse clicks to select a feature. Sometimes the number is different. If you select the *Modify* command first, then select the feature, you only use two clicks.

The thing to remember is that when you select a feature, it is surrounded with a **yellow** bounding box. When you select the part, it is surrounded with a **white** bounding box.

Modify the dimension back to 50mm.



Deselect All



Select the first extrusion feature again.



pick front face of part (don't move mouse)



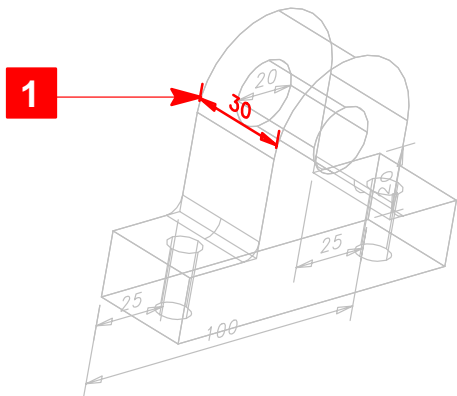
pick again



(Accept)



Show Dimensions



Modify Dimension form



Distance: 50

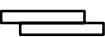


OK

Then, update twice.



Recovery Point



*File
Save*

Modify a feature's wireframe geometry1 of 8

When you create features with wireframe geometry (sometimes referred to as a sketch pad), you can display the wireframe geometry and modify it using the *Wireframe* option of the *Modify* command.

There are two menu options that are similar:

- *Quick Wireframe*
- *Wireframe*

The *Quick Wireframe* option superimposes the wireframe right on the finished part.

The *Wireframe* option rolls back the part to the state it was in when the sketch was made, which is more accurate, but might take more time.

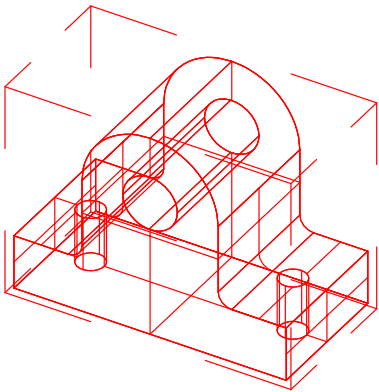
The *Wireframe* option is the only one demonstrated in this section, but you may try both.

With the wireframe section of a feature displayed, you can:

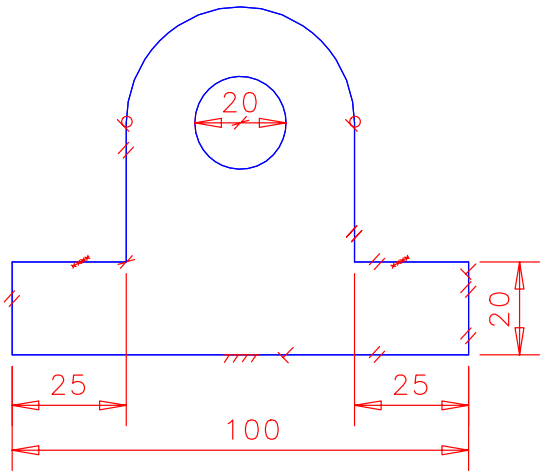
- modify dimension values
- add or delete constraints and dimensions
- add or delete curves defining the section

Modify a feature's wireframe geometry2 of 8

Select the first feature and the *Wireframe* option of *Modify*.



Wireframe



Modify a feature's wireframe geometry 3 of 8

Delete the dimensions and the parallel constraints from the two vertical lines.



1

2 shift-pick



(Done)



Yes

3

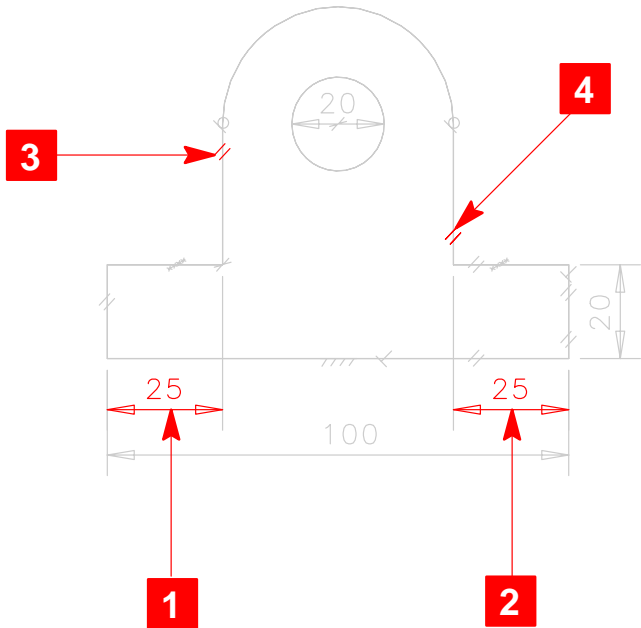
4 shift-pick



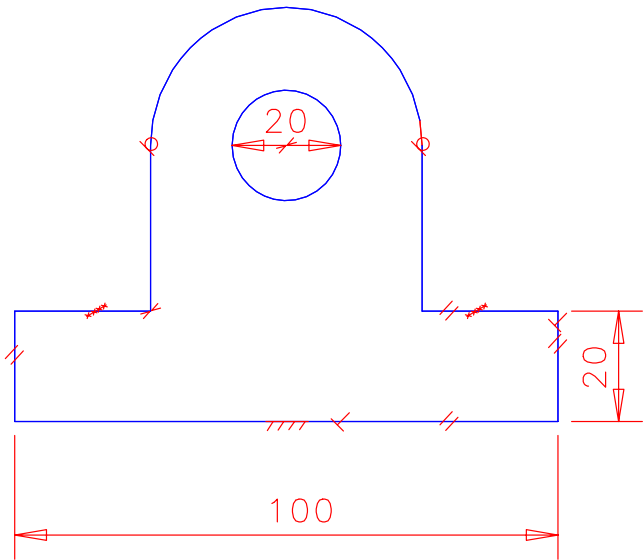
(Done)



Yes



Modify a feature's wireframe geometry 4 of 8



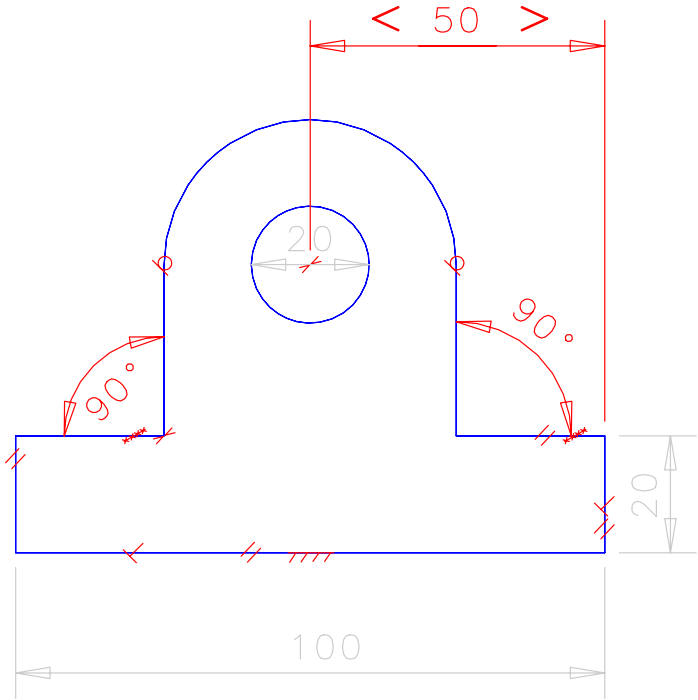
We deleted the dimensions because they were line-to-line dimensions between parallel lines. You cannot delete parallel constraints without also deleting the line-to-line dimensions.

Modify a feature's wireframe geometry 5 of 8

Add angular dimensions as shown.

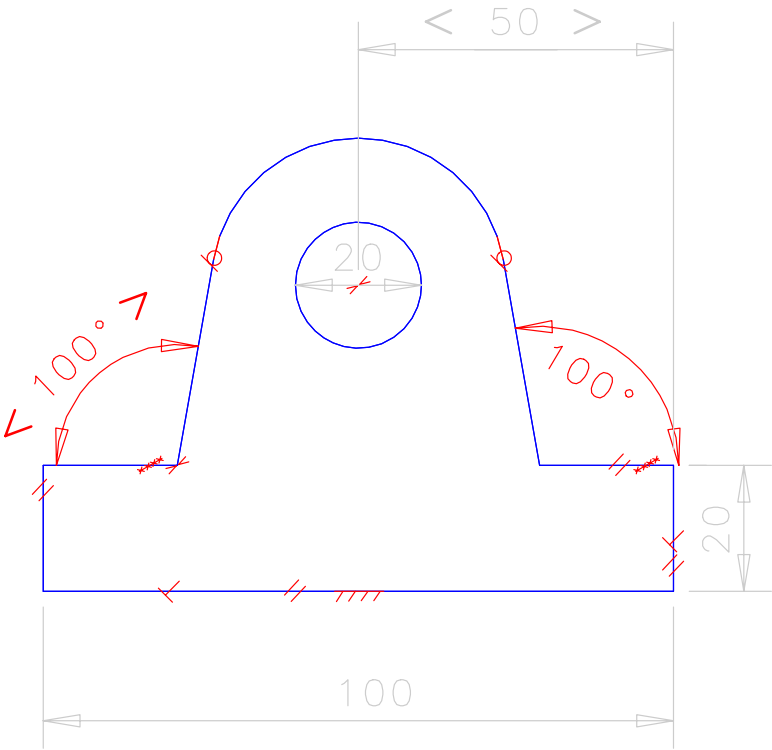
Add a point-to-point horizontal dimension as shown.

Match it to be one-half of the total width.



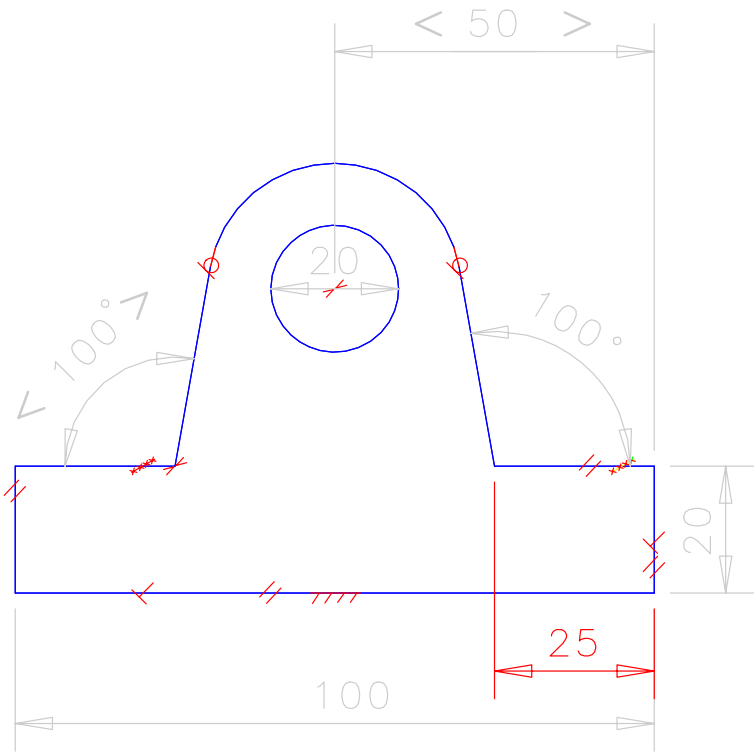
Modify a feature's wireframe geometry 6 of 8

Modify one angle to match the other, then modify the angle to 100 degrees.



Modify a feature's wireframe geometry 7 of 8

Dimension the flange width to be 25mm, as shown.

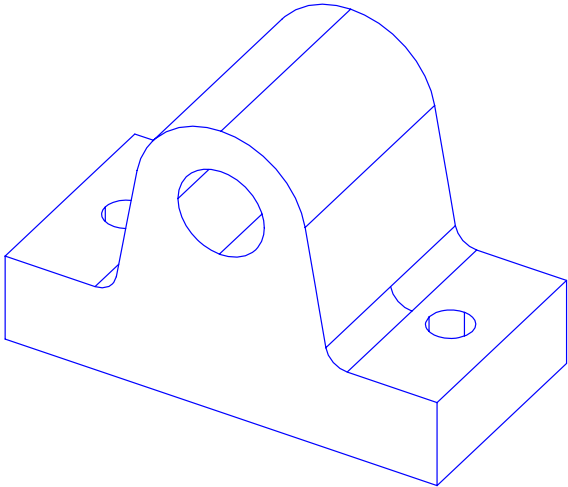


Things to notice

Due to symmetry, you need to dimension only one side.

Modify a feature's wireframe geometry 8 of 8

Update the part.

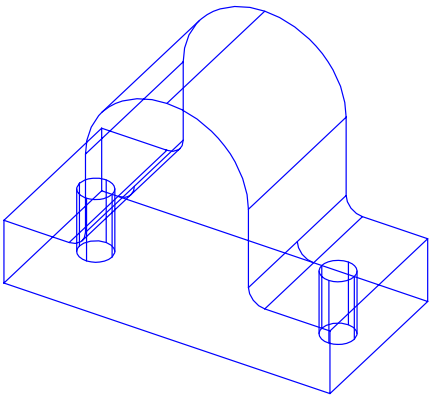
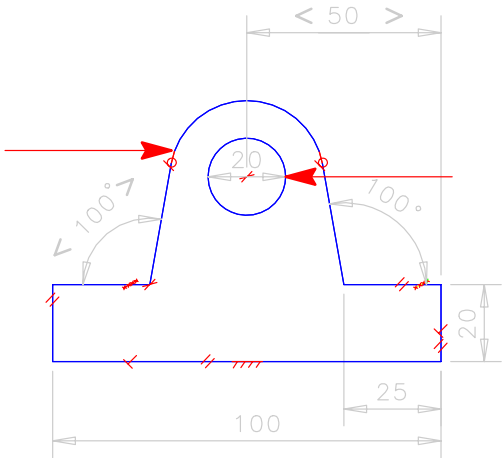


Recovery Point



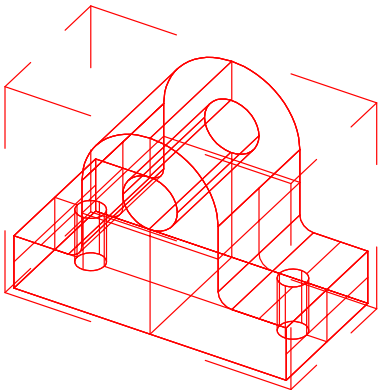
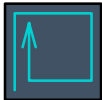
Modify a section underlying a feature1 of 5

You can modify a feature by adding or removing sections from extrude operations. In the next steps, you'll remove the section that cuts out the hole in the part. You'll then select the section again to recreate the hole.

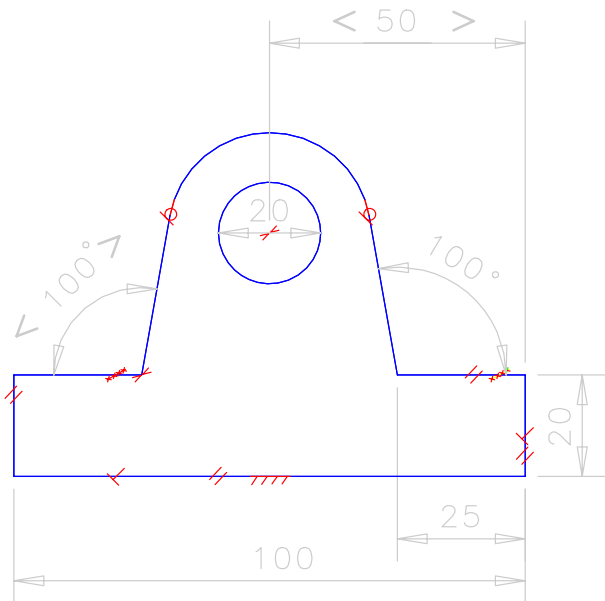


Modify a section underlying a feature2 of 5

Pick *Modify*, then the first feature.



Wireframe



Modify a section underlying a feature3 of 5

Still in *Modify*, select the section to modify. Then, deselect the circle that creates the hole.

1

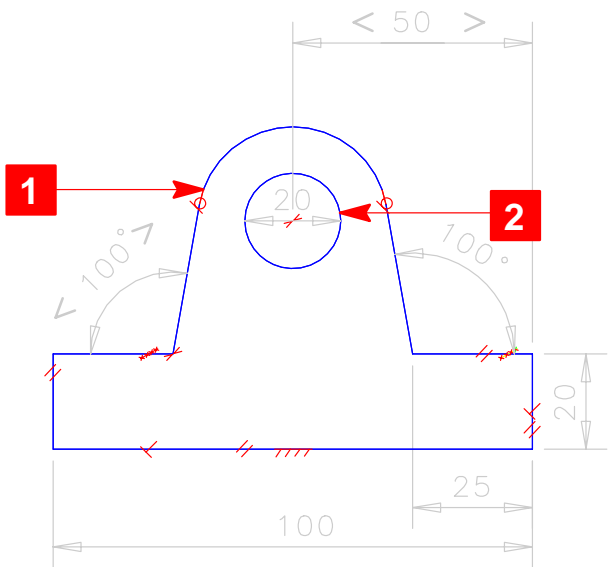


(Accept)

2

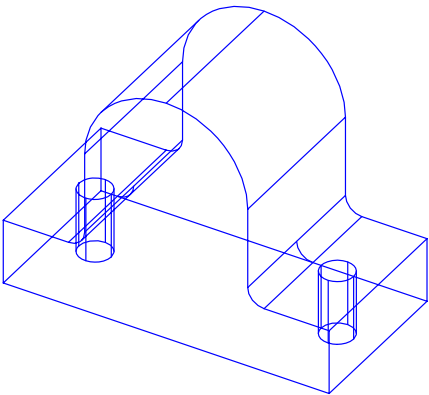


(Done)



Modify a section underlying a feature4 of 5

Update the part twice.

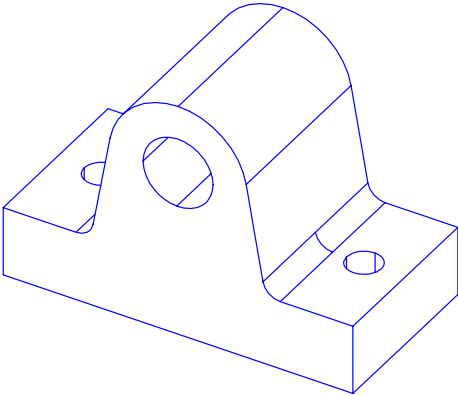
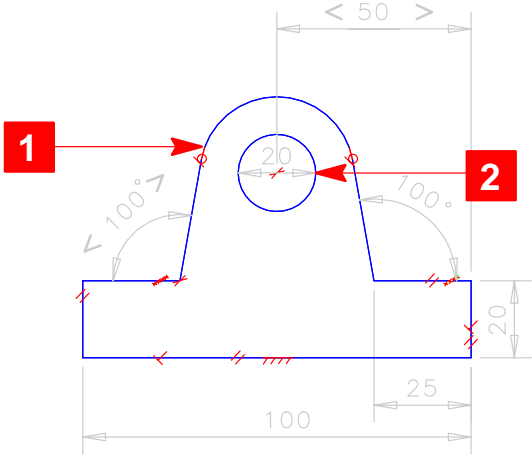


Recovery Point

 *File*
Save

Modify a section underlying a feature5 of 5

Repeat the last three steps but this time add the circle back into the section and recreate the hole.



Tutorial wrap-up

You have completed the Modifying Features tutorial.

Save the model file with this part. You'll use it in the next tutorial.